Splice: An Analytical Network Analysis Software

Thomas G. Engel, *Member, IEEE*

***Abstract*—Splice is a symbolic circuit analysis program that operates under the Windows operating system. At the present time, Splice is the only commercially available software that will perform this type of analysis. A description of the features found in Splice 1.0 and examples to illustrate them is presented. Splice works with The Design Center circuit simulator available from the MicroSim Corporation. The user adds special "analysis symbols" to the circuit schematic that instruct Splice to perform a specific type of analysis. The analysis symbols are transparent to PSpice, giving the user the option to simulate the network at any time. Splice is powered internally by the Waterloo Maple Software algebraic engine and will analyze many types of linear and linearized electrical networks consisting of inductors (with initial conditions), capacitors (with initial conditions), resistors, transformers, transistors (bipolar and field-effect), operational amplifiers, and all types of ac and dc (independent and dependent) current and voltage sources. Splice will perform branch current, nodal voltage, Thevenin reduction, transfer function, and two-port reduction analysis with options for component substitutions (both numeric and/or symbolic including zero and infinity), pole and zero calculations, sinusoidal steady-state analysis, and time domain result calculations. Potential classroom applications of Splice 1.0, as well as future enhancements of the software, are presented and discussed.**

I. Introduction

SIMPLY STATED, Splice 1.0 is a symbolic analyzer for electrical circuits and networks. It is the only commercially available software that gives exact, closed-form solutions for most electrical circuit parameters of interest in either symbolic, numeric, or mixed symbolic-numeric form. At the present time, Splice is just being introduced into the college classroom. The results of this endeavor, unfortunately, will not be known for some time. It has proven itself, however, to be a useful software tool to the various experienced engineers who are using it. Splice runs under the intuitive Windows graphical user interface. It works in conjunction with The Design Center (i.e., more commonly referred to as PSpice) [1], which many users have stated is a desirable feature. The simultaneous operation of Splice and PSpice allows the user to enter a circuit through a graphic interface and either perform a simulation (in PSpice) or obtain an exact solution (in Splice). The no-charge, freely distributable evaluation version of PSpice (version 6.1a) is shipped with Splice. Internal and operating transparently to the user, Splice is powered by the Maple V algebraic engine [2]. The user does not need the Maple V software itself, unless he wants to perform some additional mathematical manipulation of the results (e.g., calculate the maxima or minima of a Splice result). Splice will analyze all types of linear (and linearized) electrical networks, with the exception of transmission line networks, for many different types of circuit analysis. The types of analyzes includes branch current, nodal voltage, Thevenin reduction, transfer function derivation, and two-port (Y and Z matrix) parameter derivation. Splice also provides for the "other calculations" associated with any type of circuit analysis including symbolic and/or numeric component substitutions (including the values of zero and infinity), calculations of the function poles and zeros, calculations of the of the time domain results, calculation of the sinusoidal steady-state results, and the impulse and unit step response. All Splice-calculated results can be copied into other Windows programs including word processors (to be used in publications) and mathematical analysis programs (to mathematically manipulate the results further), such as Maple V.

Manuscript received September 11, 1995; revised June 5, 1996.
The author is with the CaaGE Software, Columbia, MO 65205-4005 USA.
Publisher Item Identifier S 0018-9359(96)06749-0.

Customer surveys and evaluations have shown that Splice is beneficial to all types of technical, and some nontechnical, personnel including educators, engineers, scientists, students, and technicians. Educators have found that Splice is a useful reference tool when preparing lectures and tests. Engineers, scientists, and technicians have stated that Splice has significantly reduced the time and labor involved in evaluating a problem or new idea, especially when that problem or idea is cross-disciplinary in nature (e.g., electromechanical systems). Students learning circuit and systems theory have commented that Splice provides immediate feedback as to the correctness of an answer and has significantly increased their rate of learning. The potential applications of Splice 1.0 in electrical engineering education are of primary interest and could be in the areas of circuit theory, electronics theory, project laboratories, and control and systems theory. To integrate Splice into the classroom, one strategy might be to structure the course material so as to encourage the student to try various "what-if" scenarios. This strategy would help the student to develop his or her ability to think in both an abstract and a critical manner. Splice can also be used by the student to check his or her answers, providing immediate feedback and increased rates of learning. Both strategies encourage the student to examine the mathematical form of the answer and understand the implications of that form. "What are the salient terms and components," "what terms can be neglected," and "what does this answer really show," are a few examples of the kinds of abstract, critical thinking that needs to be developed in students so that they can be productive engineers.

II. Description of Operation

The network is first created (i.e., drawn) in PSpice using various standard components. PSpice has a graphical schematic entry program that greatly speeds the creation of the electrical circuit. Next, special symbols called "analysis symbols" are added to the circuit. The Splice 1.0 software package includes

0018-9359/96$05.00 © 1996 IEEE

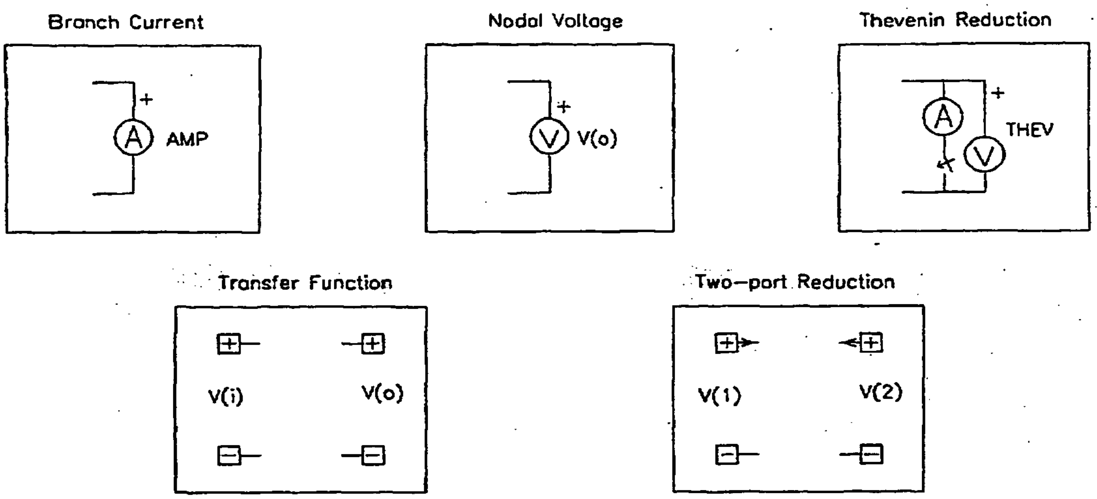

Fig. 1. Illustrating the PSpice analysis symbols for the five types of Splice circuit analysis.

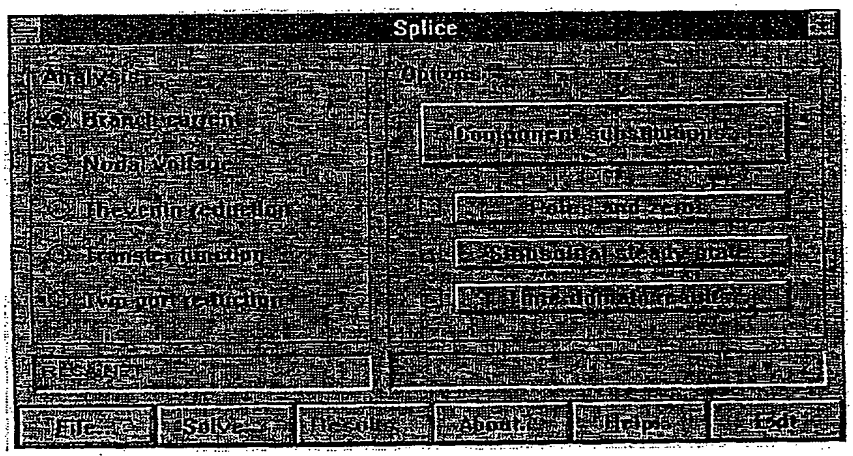

Fig. 2. Illustrating the Splice main menu and the various solution options available to the user.

analysis symbols for:

1) branch current;
2) nodal voltage;
3) Thevenin reduction (including the Thevenin voltage, impedance, and admittance);
4) transfer function derivation (including the impulse and unit-step response);
5) two-port derivation analysis (including the Z and Y two-port matrix parameters).

These symbols are shown in Fig. 1. The circuit is then saved as a netlist file, which is subsequently opened in the Splice program. Fig. 2 illustrates the Splice main menu and the various solution options available to the user. These options include:

1) component substitutions (i.e., both symbolic and numeric including zero and infinity);
2) poles and zeros calculations;
3) sinusoidal steady state calculations;
4) and time domain results.

Note that the above solutions options are not all available simultaneously and depend upon the type of analysis that is being performed.

A. Components Recognized By Splice

All linear components, with the exception of transmission lines and some linearized components, are currently recognized by Splice. These components include inductors (with initial conditions), capacitors (with initial conditions), resistors, transformers, transistors (bipolar and field-effect), operational amplifiers, and dc and ac (independent and dependent) current and voltage sources. These components are shown in Fig. 3.

B. Example Calculations

As an example of the Splice program operation, consider the circuit shown in Fig. 4. This circuit is a series connected resistor–inductor–capacitor (RLC) circuit with capacitor ini-

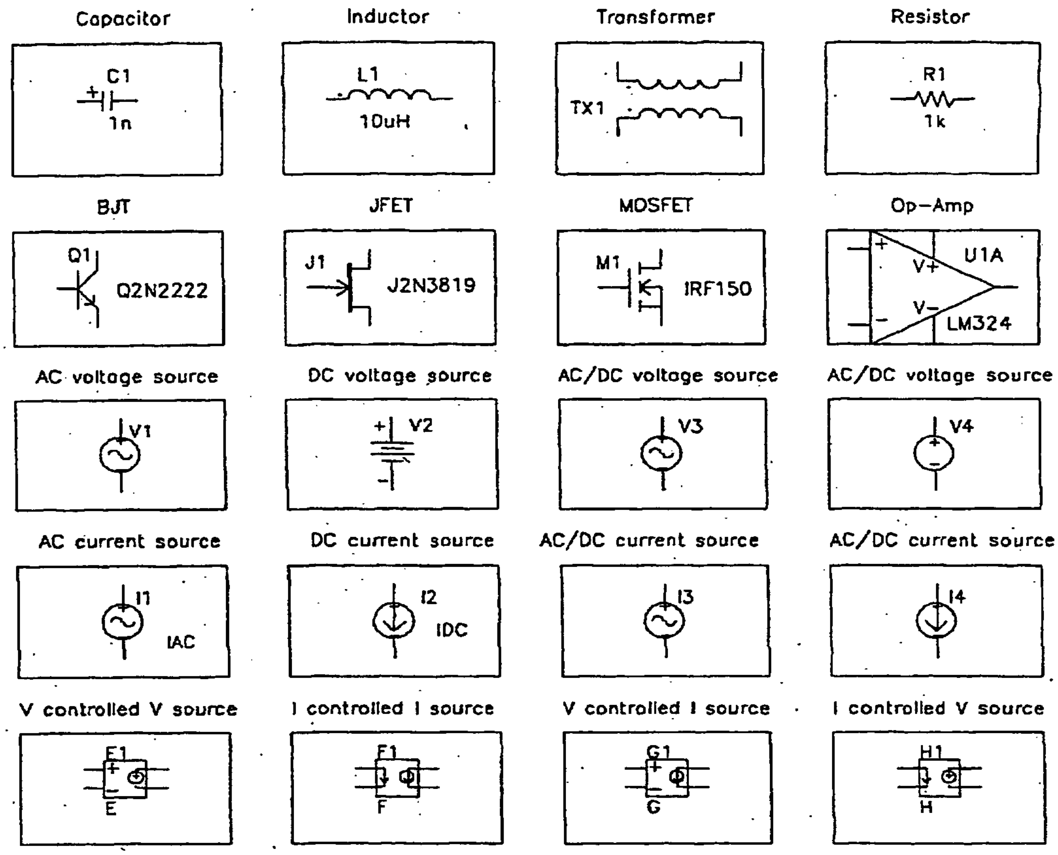

Fig. 3. Illustrating the linear and linearized components recognized by Splice.

tially charge to V_C volts and where the loop current is the desired solution. Although this circuit is a simple example, it will serve to illustrate the operation of Splice. Importing the netlist of this circuit into Splice and specifying that the time domain results are to be calculated, the series RLC circuit current is given as

$$i_{branch\_t} = 2V_C\sqrt{C}\,e^{-(Rt/2L)}\sin\left(\frac{\sqrt{4L-R^2Ct}}{2L\sqrt{C}}\right) \cdot \frac{1}{\sqrt{4L-R^2C}}. \quad (1)$$

It is to be noted that (1) was exported from Splice and directly into the word processor (i.e., AmiPro in this case) in LaTeX format, thereby preserving the form of the equation. Besides the LaTeX format, Splice also has features that permit the user to copy the results in linear and plain text (ASCII) format. LaTeX format is supported by most word processors, while the linear format allows the user to copy the results into mathematical analysis programs (e.g., Maple V) for further mathematical manipulation. The plain text format is provided for all the other cases.

Another good example of Splice operation is the transformer circuit shown in Fig. 5. In this circuit, the desired calculation is the Thevenin reduction at a particular node pair (see Fig. 5).

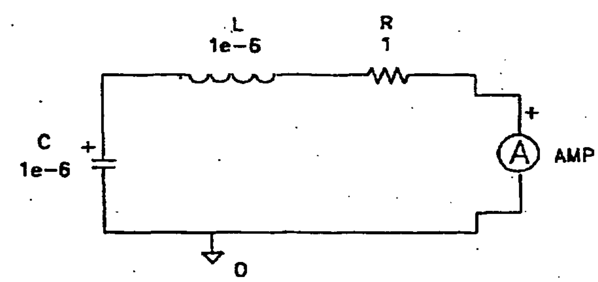

Fig. 4. A series RLC circuit where the branch current is the desired Splice calculation.

The point here is that while most circuits are simple enough to allow the user to perform a series-parallel combination of the circuit impedances to arrive at the Thevenin impedance, the transformer coupling in the circuit of Fig. 5 complicates that type of approach. Substituting the usual three-inductor, "T" equivalent circuit of the transformer into the circuit allows the user to combine the impedances with series-parallel combinations, but produces an incorrect final answer. The correct Thevenin impedance for this circuit is

$$z_{\text{Thevenin}} = \frac{(4+3s^2CL)sL}{s^4C^2L^2+3s^2CL+2}. \quad (2)$$

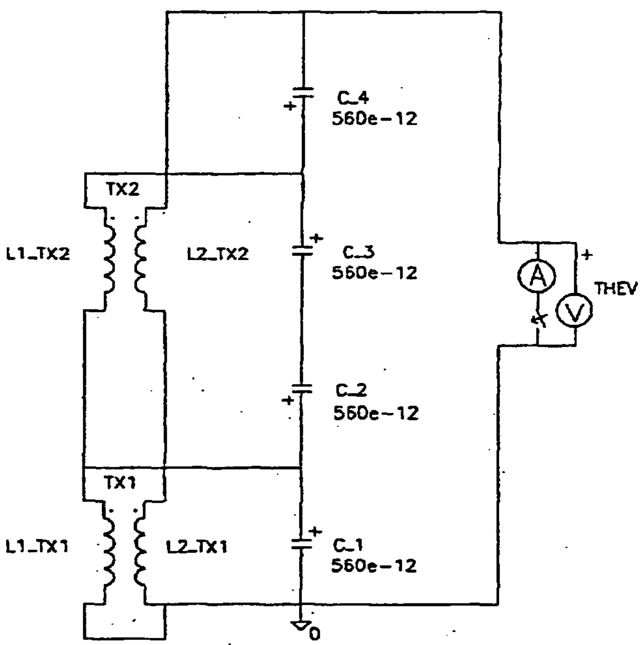

Fig. 5. A transformer circuit where the Thevenin reduction is the desired Splice calculation and where $C_1 = C_2 = C_3 = C_4 = C$; $L1_{TX1} = L2_{TX1} = L2_{TX2} = L$; and the transformer coupling values $K_{TX1} = K_{TX2} = 1$.

The CD-ROM that accompanies this manuscript contains a demonstration version of the Splice 1.0 software that contains more detailed information of the operation and features of Splice. The demonstration program also contains other example calculations for the five different types of circuit analysis that it will perform. Examples of pasting the Splice-calculated results into word processing software and mathematical analysis software are also provided.

C. Course Curriculum Philosophy

It is this writer's opinion that the primary objective of electrical engineering education should be to provide students with the tools that they will need to begin their careers and to continue learning in those careers. Given these tools, the student should become a successful engineer. Tools (e.g., hammers, chain saws, nail guns, etc.) that are improperly used, however, can cause a good deal of injury. Splice is also a (software) tool that can be improperly applied in that its use should be more than providing the students with a piece of software that will calculate an answer for them. Splice could be used in electrical engineering courses as a tool for providing immediate, positive feedback to the student as he or she is learning the principles of circuit theory, electronics, or systems theory. It should be noted that positive, immediate feedback is an important part of a rapid, efficient learning process [3], [4] for a large majority of students. As the students learn how to perform a given type of analysis, Splice can be used to help them develop a sense for the correct answer and to learn how to interpret the functional form of an answer. In higher-level courses, Splice could be used as both a reference tool and an optimization tool. If the instructor accepts the educational objective stated above, then the implementation of Splice into the classroom could be accomplished with a two-step approach: teach the basic concepts and reinforce those concepts with examples.

Basic concepts in electrical engineering are powerful tools and should be thoroughly understood by the student at the earliest stage in his or her development. The finer points of engineering emerge from the basic concepts and can be taught and learned more efficiently once the student thoroughly understands the basic concepts. Using this approach to learning, students should be taught to develop the ability to reduce a new problem into its basic elements so that the problem can then be solved. For example, in a beginning circuits course, the operation of a transformer and its two-port "T" equivalent circuit (using three inductors) would be considered to be a basic concept. Splice could be used in these types of problems to serve as a computerized check of the student's answer. The immediate feedback afforded by Splice would allow the student to learn at a faster rate.

Reinforcing the basic concepts with examples will embed those concepts and allow the student to proceed to the other, finer points in electrical engineering and is similar to the teaching philosophy of project laboratories in that abstract theoretical concepts are reinforced with practical, real-world measurements and results. Using the operation of a transformer as a basic concept, a finer point of engineering would be to have the student specify the transformer inductances and capacitances that would allow it to function as a pulse transformer, a power transformer, or a high-frequency transformer. Another finer point would be to show that the "T" equivalent circuit is not an accurate model of the transformer for all circuit configurations. Splice would be used in these types of problems to calculate the impulse, or step, response of the transformer to determine the effects of various component changes. With reinforcement by example, students develop a better understanding of the basic concepts, have a higher degree of retention, and are able to progress to high levels of learning.

D. Comparison to Other Symbolic Circuit Analyzers

In comparison to other symbolic circuit analyzers, Splice is the only analyzer that includes the majority of features that are of primary importance to the electrical engineer. The handful of other available circuit analyzers offer only a limited coverage of analysis functions [5], [6]. For example, most of the other analyzers offer only transfer function analysis and/or nodal voltage analysis. Splice offers a full range of analysis functions including branch current, nodal voltage, Thevenin reduction, transfer function derivation, and two-port parameter derivation. The solutions options available with other analyzers also appears to be inadequate. While most of the other analyzers can express the results in the frequency domain, with some being able to calculate the sinusoidal steady-state results and perform pole–zero calculations, Splice can additionally perform the useful functions of time domain

results representations, impulse and unit-step response calculations, and component substitutions of zero and infinity. An added feature found in Splice and few of the other analyzers is its graphical schematic capabilities. Therefore, it appears that Splice is the only full-featured symbolic circuit analyzer that can perform the majority of analysis functions with the useful solutions options needed by most electrical engineering applications.

III. Conclusion and Future Enhancements

Splice 1.0 is the only commercially available symbolic circuit analyzer that performs the types of analysis (i.e., branch current, nodal voltage, Thevenin reduction, transfer function derivation, and two-port parameter derivation) with the solution options (i.e., pole–zero calculations, sinusoidal steady-state representation, time domain representation, and numeric, symbolic, and mixed numeric/symbolic component substitutions) that are of primary interest to the electrical engineer. It appears to be a useful software tool for both the student and the experienced electrical engineer. Its features permit the user to explore a wide variety of electrical circuit analysis and to export the results into other word processors (for publications) and mathematical analysis programs (for extended mathematical analysis). If used properly, Splice could be a useful educational tool to help teach students the basic concepts in electrical engineering and help them explore the more intricate details of those basic concepts. Future software enhancements will include a stand-alone version that does not require the PSpice schematic entry program and the development of versions for other operating systems. Analysis support for transmission lines and possibly nonlinear components will need to be added to the software to increase its functionality. Support for other types of analysis, including circuit quality factor and stability analysis, will also be added. Future investigations will explore and report the suitability of Splice in the classroom education of electrical engineering students. Employing Splice in the next-generation multimedia educational tools could prove to be useful and will also be explored.

References

[1] *The Design Center, ver. 6.1a*, MicroSim Corp., Irvine, CA.

[2] *Maple V, release 3*, Waterloo Maple Software, Waterloo, Ontario, Canada.

[3] R. Viau and R. E. Clark, "Guidelines from research: Feedback during training," *J. Performance and Instruction*, vol. 26, no. 3, pp. 6–7, 1987.

[4] R. S. Lysakowski and H. J. Walberg, "Classroom reinforcement and learning: A quantitative synthesis," *J. Educational Research*, vol. 75, no. 2, pp. 69–77, 1981.

[5] G. Gielen *et al.*, "Symbolic analysis methods and applications for analog circuits: A tutorial overview," *Proc. IEEE*, vol. 82, no. 2, pp. 287–304, 1994.

[6] L. P. Huelsman, "Symbolic analysis—A tool for teaching undergraduate circuit theory," *IEEE Trans. Educ.*, vol. 39, no. 2, pp. 243–250, 1996.

Thomas G. Engel (S'87–M'90) was born in 1959 in Houston, TX. He received the Ph.D. degree in electrical engineering from Texas Tech University, Lubbock, TX, in December 1990. His dissertation topic was insulator degradation produced by high-current surface discharges.

From 1992 to 1995, he was a Research Assistant Professor in the Pulsed Power Laboratory of Texas Tech University, where he continued his research in insulator physics and explored industrial applications of pulsed power technology. He became an Assistant Professor of Electrical Engineering in August, 1995, at The University of Missouri at Columbia. His present interests and specialties include insulator physics, plasma physics, and circuits and systems. He has published 40 journal and conference articles in his fields of specialization and served as co-editor (with H. H. Cohen and F. M. Leh) for *SPEAR II: High Power Space Insulation* (Lubbock, TX: Texas Tech University Press, 1995).

Dr. Engel is a member of Tau Beta Pi, Eta Kappa Nu, Sigma Xi, and TTEEA (Texas Tech Electrical Engineering Association).